AIAA AVIATION Forum
June 15-19, 2020, VIRTUAL EVENT
AIAA AVIATION 2020 FORUM

10.2514/6.2020-2772

Requirements and Challenges for CFD Validation within the High-Lift Common Research Model Ecosystem

Adam M. Clark[1] and Jeffrey P. Slotnick[2]
The Boeing Company, Seattle, Washington, 98124

Nigel Taylor[3]
MBDA UK Ltd., Filton, Bristol, BS34 7QW, UK

Christopher L. Rumsey[4]
NASA Langley Research Center, Hampton, VA, 23681

The High-Lift Common Research Model (CRM-HL) ecosystem is envisioned to become an industry standard set of test cases for high-lift aerodynamics prediction, with data generated providing a strong foundation for CFD validation. Since its original development, preliminary wind tunnel tests have been conducted to further understand and configure the geometry, and several more are in the planning phases. It is anticipated that the data generated in these tests will be used extensively by the CFD community for research, development, and validation of new techniques that yield more accurate results. To derive the maximum benefit from planned wind tunnel testing, data acquisition will be focused in several key areas to address specific shortcomings in CFD predictive capabilities. Significant challenges currently exist in areas including, but not limited to, characterizing high-lift flow phenomena, quantifying effects due to configuration changes, understanding wind tunnel and model installation effects, and establishing data uncertainty. This paper will discuss the current state-of-the-art in high-lift CFD prediction, key limitations of current CFD methods and tools, establishing a validation dialog between test and CFD practitioners, and the flow phenomena of highest interest, in an effort to begin to address these challenges. To this end, it is expected that the data generated from CRM-HL tests will form a strong foundation for future predictive capability.

1. Introduction and Overview

Numerical prediction of the aerodynamic performance of subsonic transport aircraft in landing or takeoff (high-lift) configuration is particularly challenging for the current generation of Reynolds-Averaged Navier-Stokes (RANS) computational fluid dynamics (CFD) tools. High-lift flowfield prediction is driven, in large part, by turbulent flow separation and the control of both its development and impact on aerodynamic performance, an area where there are many known deficiencies in numerical modeling [1]. Lack of capable and efficient predictive tools, coupled with the sheer number of configurations evaluated over the course of an airplane development program, requires that the wind tunnel remain the tool of choice for practicing aerodynamicists tasked with characterizing high-lift performance. However, the modern wind tunnel comes with its own set of limitations that can cause observed variations in flow phenomena that may not be well understood or that may be difficult to characterize. These include Reynolds number

---

[1] Aerodynamics Engineer, Flight Sciences.

[2] Boeing Technical Fellow, Flight Sciences, Associate Fellow AIAA.

[3] Capability Leader, Aerodynamic Tools and Methods, AIAA Associate Fellow.

[4] Senior Research Scientist, Computational AeroSciences Branch. Fellow AIAA.

effects, the influence of constraints imposed by the wind tunnel and model mounting systems, and differences between flight and wind tunnel geometry caused by design and manufacturing considerations related to the wind tunnel models. Moreover, design and development of an efficient slotted high-lift system for a modern commercial airliner requires careful consideration of a number of complex interacting factors, including the balance of contributions made by the leading edge device, main element and trailing-edge flaps in the various phases on takeoff and landing. In this context, consideration of a full vehicle configuration is necessary for advancing the predictive capabilities of computational tools.

The High-Lift Common Research Model (CRM-HL) defines a modern, industrially-representative, precompetitive high-lift geometry openly available in the public domain [2]. Configurations based on this geometry are envisioned to have many uses within a broader CRM-HL "ecosystem," one of which is to provide for high-quality wind tunnel models and testing to generate a comprehensive set of data for high-lift CFD validation. Validation utilizing the open CRM-HL geometry will be realized through public efforts such as the AIAA CFD High-Lift Prediction Workshops (HLPW), as well as through internal commercial efforts within the aerospace industry.

As the CRM-HL ecosystem continues to expand in scope to involve multiple partners testing models in multiple wind tunnel facilities over several years, there is a critical need to develop test plans that are based on comprehensive and unified requirements that will drive and support computational validation efforts, as well as ensure consistency in acquiring high-quality test data. These requirements will be derived from a variety of motivations including increased understanding of key aerodynamic aspects unique to high-lift configurations, increased understanding of uncertainty in the context of high-lift CFD and wind tunnel testing, and better quantification of the effects of wind tunnel mounting systems, among others. As an example, the CRM-HL ecosystem creates a unique opportunity to test identical configurations, and in some cases identical models, across multiple facilities with differing mounting configurations. If data are collected in a deliberate and systematic fashion, they can be used to increase the confidence in predictive tools.

CFD continues to garner increased interest in the aerospace industry not only for design purposes, but also to support airplane certification efforts. Despite the advantages associated with the use of the wind tunnel, the limitations discussed above, as well as long lead times associated with model design and fabrication, provide an opportunity for numerical analysis to impact Certification by Analysis (CbA) [3]. However, to be effective for CbA applications, CFD validation requires dedicated efforts to collect relevant ground test data that can be reliably correlated with flight test data. It is within the context of both aerodynamic design and CbA that the CRM-HL ecosystem has been established.

This paper represents an initial effort to highlight and characterize the challenges associated with validating high-lift CFD predictions, and begin to outline requirements for how they can be addressed using the CRM-HL ecosystem in the context of supporting current aspirations for improving confidence in CFD predictions. These requirements have been captured through discussion within industry, among CRM-HL ecosystem partners, and through public venues such as AIAA conferences. However, as they are presented here in outline form, there is not sufficient detail in the requirements to adequately define wind tunnel models, instrumentation, specific test matrices, or CFD studies. Rather, these requirements are collected to provide guidance for future integrated testing and simulation within the CRM-HL ecosystem, and are expected to evolve as the community understanding of this high-lift configuration increases.

2. Current State of CFD

In 2010, the AIAA HLPW, an open international workshop series, was established to help advance the state-of-the-art in predicting high-lift flows. To date, three workshops have been held; more are planned for the future. HLPW-1 [4, 5] focused on the three-element NASA Trapezoidal Wing configuration [6, 7]. HLPW-2 [8] made use of the DLR-F11 configuration [9]. Finally, HLPW-3 [10] employed both a simplified version of the High-Lift version of the Common Research Model (CRM-HL) [2] and the JAXA Standard Model [11].

All workshops have facilitated both general and statistical analyses of the computational results supplied by the participants. A considerable amount has been learned from them to date. Insofar as the workshops reflect the current state-of-the-art, both retrospective and forward-looking perspectives on their main conclusions are provided below.

Throughout the rest of the paper, we refer to verification and validation. These two terms have each come to mean something very specific within the context of CFD [12]. Verification is the process of establishing the correctness of the coding and computational procedures in a CFD code and the way they have been applied. Validation involves assessing the accuracy of a CFD result by comparison with physical measurements. In verification, the relationship of the simulation to the real world is not the focus; the important thing is accurately coding and solving the equations as intended. In validation, the relationship between computation and the real world is the focus; i.e., assessing how well the model equations represent the physical reality.

1. Retrospective

Key learnings from the past workshops in two specific areas – code verification and the CFD modeling assumptions used in the prediction of high-lift configurations – are discussed here to highlight the particular difficulties in predicting high-lift flows, but also to hint at specific requirements for future testing and CFD simulation necessary for improved predictive capabilities.

Code Verification

One of the overarching problems observed during the workshops was the large observed variation in the computational results supplied by participants, particularly near maximum lift ($C_{L,\,max}$) conditions, as depicted in Figure 1. Although this figure includes outliers whose solutions are clearly questionable, the spread is quite large even if the outliers are ignored. Even for a specific set of modeling assumptions (e.g., a particular choice of turbulence model), results displayed unaccountably large variations. Apart from other issues, these inconsistencies made drawing definitive conclusions about the suitability of the CFD codes, or the practices surrounding their use, difficult. Over the progression of the workshops, there was growing statistical evidence to suggest that the variation in the computational results were reduced at low angles-of-attack, but no improvement was evident near aerodynamic stall.

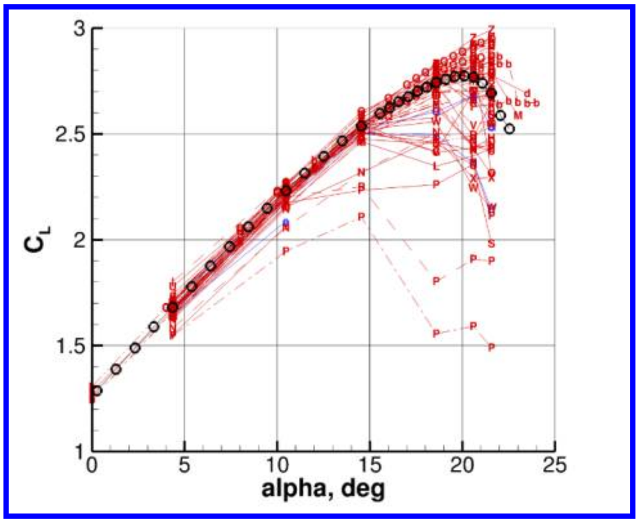

Figure 1: C_L prediction scatter from HLPW-3 [10]

The lack of consistency observed across datasets arises from a variety of sources, including the use of insufficient mesh resolution, disparities or errors in the implementation of the governing equations, and differences in the level of iterative convergence. Evidence of the influence of differences in local user practice when running the same code was also identified [13]. Another frequent issue among participants was the identification of multiple solutions on the same grid, often associated with separation in line with leading edge brackets. As a result of the diversity of factors involved, and the difficulties of isolating them systematically in a workshop context, the specific reasons for the observed inconsistencies remain unclear.

In an attempt to begin to address this challenge, participants of HLPW-2 and HLPW-3 were asked to complete RANS code verification exercises. Although results using different turbulence models were submitted, only those based on the standard implementation of the Spalart-Allmaras turbulence model [14, 15] were utilized in the workshop analysis. Even though the verification scenario was two-dimensional in HLPW-3, only about 30% of the code results were able to match established computational results for the verification case, suggesting that at least some of the observed inconsistencies in workshop results were due to implementation errors or inconsistent coding practices.

CFD Modeling Assumptions

Occasionally, submitted results from CFD simulations have exhibited reasonable agreement with data acquired from wind tunnel tests near maximum lift, in terms of lift coefficient. However, in many of these cases, such agreement has been shown to be fortuitous. For instance, in HLPW-3, most of the RANS computational results acquired near the stall condition showed very large regions of reversed flow outboard over the main wing, but little or no separation near the wing-root juncture. In contrast, oil flow visualizations acquired during wind tunnel testing showed significantly different separation patterns. An example from Ito et al. [16] is shown in Figure 2. The left side of the figure is an oil flow photograph taken near stall, with a large region of wing-root separation and only relatively small separation outboard on the wing. The right figure shows skin friction contours from RANS CFD, where the solution indicates very little inboard separation, along with larger outboard separation.

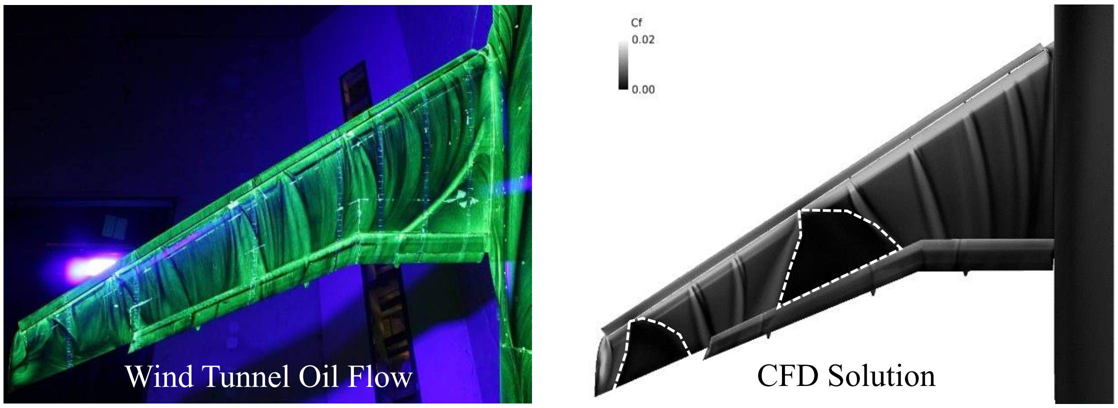

Figure 2: HLPW-3 [10] oil flow (left) compared with RANS results on a fine (right) mesh; Figure taken from Ito et al. [16], used with permission.

Pressure coefficient data submitted to the HLPW were useful in highlighting specific areas where CFD failed to adequately capture the flow physics observed during wind tunnel testing. For example, the data from HLPW-2 showed that CFD predictions tended to exhibit the greatest inconsistencies and discrepancies from physical measurements on the flap (where the flow is separated), and outboard (near the wing tip). Figures 3 and 4 show examples at two span stations. Near midspan (Figure 3), the slat and main element (left and center figures) are predicted reasonably well by CFD, but the separated flap (right figure) shows greater variability between results, and deviation from the wind tunnel data. At the outboard span station (Figure 4), CFD results on all three elements are highly variable, and inconsistent with wind tunnel data.

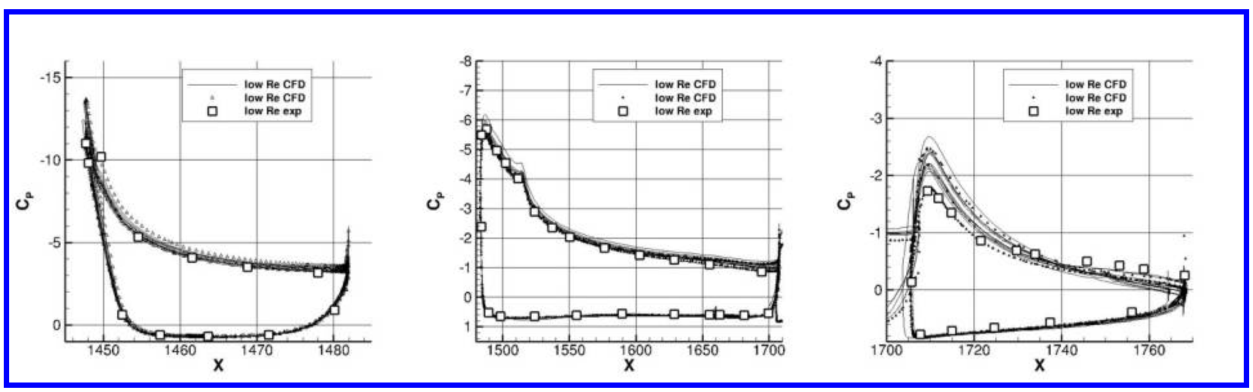

Figure 3: Midspan Pressure coefficient distributions on the slat (left), main element (center), and flap (right) near $C_{L,\,max}$ from HLPW-2 [8]

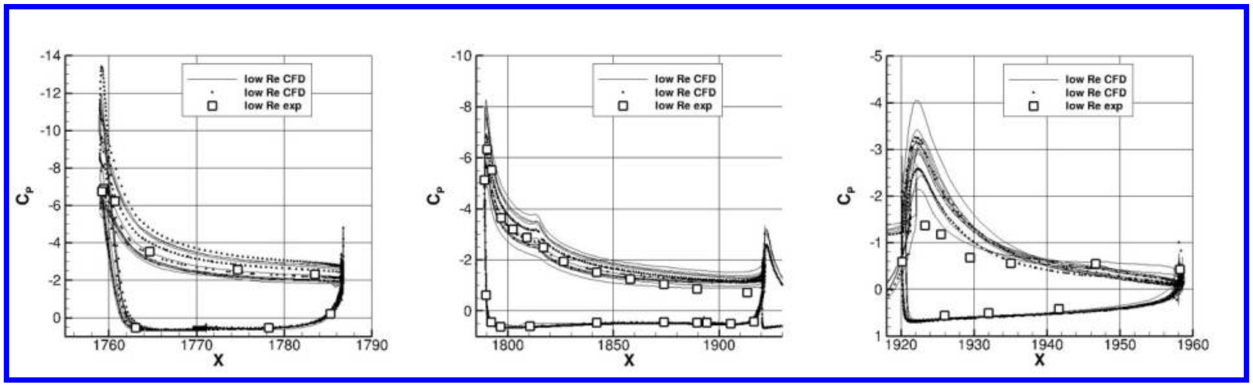

Figure 4: Wingtip Pressure coefficient distributions on the slat (left), main element (center), and flap (right) near $C_{L,\,max}$ from HLPW-2 [8]

The reasons for the apparent poor representation of the flow near maximum lift in the CFD simulations are very difficult to diagnose. However, apart from possible inconsistencies stemming from lack of code verification as discussed above, there are a number of reasons that likely contributed to the observed differences between the workshop CFD predictions and wind tunnel data.

First, workshop participants were asked to run the CFD simulations in "free-air" to compare to wind tunnel wall-corrected test data, as opposed to simulating the physical in-tunnel semispan model set-up and comparing directly with uncorrected test data. However, the interactions of the model with the floor boundary layer, together with the three-dimensional variations in wall-induced upwash and blockage, all likely play a major role in the exact nature of the flow development over the wing.

Second, geometric differences between the computational model and the model tested in the wind tunnel were not fully accounted for. In the workshops, the Outer Mold Line (OML) supplied to participants included various geometrical simplifications. Some of these simplifications removed features known to affect the flow (e.g., using a configuration without slat or flap brackets). Other approximations were made because required information was not readily available. For instance, even though modeling of aeroelastic deformation due to wind tunnel loads may be important, CFD simulations typically do not account for it because the model deformation information is often limited or not available at all, and the methods to incorporate deformation data into the geometry are often immature. While the precise consequences of such simplifications remain unclear, evidence of the potential importance of including smaller geometric details in CFD simulations is suggested from data analysis in HLPW-2 [8], where pressure tube bundles installed alongside slat brackets on the wind-tunnel model appeared to exert a significant influence on the nature of the flow development predicted near $C_{L,\,max}$. An example is shown in Figure 5, where one of the workshop participants ran CFD both with and without pressure tube bundles modeled.

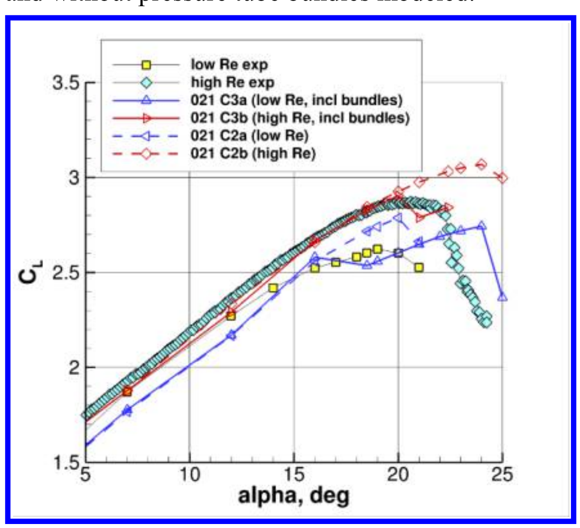

Figure 5: Effect of including pressure tube bundles on lift predictions at two different Reynolds numbers near $C_{L,\,max}$, from HLPW-2 [8]

Lastly, known simplifications in flow modeling assumptions used by the HLPW participants constitute further potential reasons for the differences observed between the CFD predictions and corrected wind tunnel data. For instance, a standard modeling assumption typically used in production RANS CFD codes is the Boussinesq hypothesis that underpins the turbulence modeling approach implemented in these tools. Even with the inclusion of terms intended to correct for observed deficiencies in their ability to model corner and rotational flows, for example, the extent to which such models are able to capture the flow physics encountered under high-lift conditions remains to be seen. There is increasing evidence that capturing the interactions of large-scale turbulent flow structures is important to accurately model significant flow separation. CFD simulation results obtained using eddy-resolving methods seem to capture flow features that were more representative of those observed in wind tunnel testing. For example, see König et al. [17], who employed a Lattice-Boltzmann methodology with similar characteristics to a very large-eddy simulation.

To date, comparisons between CFD predictions and wind tunnel measurements performed during the workshops have been limited primarily to overall forces and moments (e.g., lift, drag, and pitching moment), discrete surface pressures, and oil-flow data. With the exception of the limited velocity profile data provided in HLPW-2, detailed off-surface data measurements, critically important in understanding the complex interactions of the flow on fully-configured high-lift airframes, have not yet been widely available.

2. Forward-looking perspective

One of the general observations voiced during the review of the consolidated HLPW-3 results was the lack of discernable progress that had been made since HLPW-1 (a period of approximately seven years) regarding increased confidence in the generation of accurate CFD simulations near maximum lift. The difficulty in precisely knowing how to rectify this situation is a key motivator in promoting a more systematic approach to the high-lift CFD validation challenge. This will involve breaking down the overall goal of predicting high-lift aerodynamic characteristics into smaller, more focused challenges, from which conclusions may – hopefully – more readily be drawn.

By building on the experience gained during the workshops to date, together with the wealth of insights that have been accumulated elsewhere, the material below identifies some of the key areas that will need to be addressed in order to make progress. Below, we consider potential paths forward based principally on the concepts of CFD verification and validation to lay a foundation on which to accelerate our collective learnings. This approach better acknowledges the current state-of-the-art, yet does not attempt to prescribe a detailed or definitive solution strategy.

Verification

While this paper is focused on the requirements for CFD validation, further concerted effort is required in improving the current state of CFD verification. Verification is typically divided into two separate parts: code verification and solution verification [12]. As stated previously, code verification is the process of establishing the correctness of the computer code itself; i.e., insuring that the equations are coded as intended. Solution verification begins with the assumption that code verification has been successfully carried out. It then strives to ensure that the numerical errors in a given computational solution are reduced sufficiently to obtain the desired quantity of interest within a predefined level of accuracy. Further improvements in both code verification and solution verification are necessary before significant progress on validation can occur. This will require active engagement from a wide range of stakeholders, from software developers to those responsible for developing recommended practices among CFD users. The discussions below address one of the most significant and widely recognized sources of solution verification error: discretization error associated with inadequate resolution of the flow. Even for relatively simple 3-D aircraft configurations, this type of error appears to be quite significant [18].

We hope to build upon the experiences gained via HLPW-3, which was organized in parallel with the first Geometry and Mesh Generation Workshop (GMGW-1). On this occasion, data exchange between the workshops was largely conducted in series: a set of meshing guidelines was supplied to the GMGW organizing committee, who provided a suite of mesh families that could be used by HLPW participants. For future workshops, more interactive mesh/solver team-based approaches are being considered to facilitate more informed decision-making with regard to mesh generation and to provide mechanisms by which meshing guidelines can be both challenged and improved.

Mesh adaptation represents a much tighter interaction between mesh generation and flow solution, and its development remains a high priority. However, a key unanswered question is the choice of metric(s) used to drive the adaptation, particularly for high-lift flows. As is well known in the fundamental research community, the numerical approaches used to facilitate iterative convergence in central and upwind schemes (artificial viscosity and limiters, respectively) can introduce substantial dissipative losses in the vicinity of vortex cores [19]. In spite of this, there is currently limited understanding of how contemporary mesh adaptation schemes respond to this type of effect. Furthermore, users may be unaware of the existence or extent of artificial numerical effects present in their computed flows, or of the influence they have on the ability to demonstrate convergence to a continuum solution [20]. As a result, how and where flow solutions should be monitored, using either fixed or adapted meshes, remains an open question.

The points made above refer to both steady and unsteady flow simulations. The verification of unsteady flow simulations, especially those attempting to resolve rather than model turbulent structures, introduces a number of additional challenges. Aside from the requirements to demonstrate both spatial and temporal convergence, the additional computational complexity and increased cost makes their verification considerably more difficult compared to steady-state RANS. Another issue is a consequence of the dependency of the physical modeling assumptions being invoked on the local mesh size. This renders conventional approaches to demonstrating mesh convergence insufficient. The development of robust and efficient means for demonstrating solution verification for representatively complex, unsteady high-lift flows is therefore a high priority.

Validation

Once code and solution verification are more adequately addressed, it will be possible to be more assertive about the outcomes of comparisons made between the computational results and physical measurements, an essential requirement for CFD validation. As described in Section 3, such comparisons will rely heavily on physical measurements acquired during wind tunnel testing. Note that the task of assessing the validity of CFD predictions for full-scale flight is beyond the scope of this paper.

The importance of adopting a systematic approach toward CFD capability development will be as important for validation as it is for verification. To this end, it will be valuable to break down the problem of predicting high-lift aerodynamics into a set of more focused activities, each of which contributes to an improved understanding of the overall challenge. Doing so offers improved prospects for identifying a clear path for making progressive, incremental improvements to establish a truly predictive CFD capability. Key to this strategy will be the ability to identify and model the physical mechanisms governing flow development over complex, multielement high-lift configurations, and:

- How these mechanisms develop with increasing angle of attack
- How they are altered by changing the rigging of the high-lift devices (i.e., leading-edge slats and trailing-edge flaps)
- How they are influenced by the wind tunnel facility and model manufacturing constraints

To quantify these effects, further effort will be required to obtain:

- A detailed understanding (acquired using both computational and physical investigations) of isolated flow phenomena, such as vortex shedding and boundary layer transition, and the nature of their interactions with other phenomena encountered on high-lift configurations – such as the confluence of boundary layers and their interaction with vortices shed from chines, slat brackets.
- A detailed characterization of the effects associated with the systematic variation of key governing parameters – including angle of attack and slat and flap rigging. In addition to on- and off-body flow visualization data, this characterization should include a breakdown of all aerodynamic forces and moments acting on each of the principal components of the high-lift system (i.e., slats, main wing, and flaps), and computing forces as a function of span on the wing (spanload).

It is expected that studies utilizing simplified airframe configurations, and/or partial builds of a fully-configured wind tunnel model of a high-lift system will be crucial to accelerating learning. Further details of the phenomena of most immediate interest are presented in Section 4.

3. High-Lift Common Research Model Ecosystem

A key outcome from the past decade of high-lift prediction workshops is the crucial need to acquire higher-quality wind tunnel data to accelerate progress in understanding high-lift flow physics, and predicting the impact of that flow physics on the aerodynamics of full high-lift configurations. To this end, a community-sourced set of resources available from within a broad collaborative environment or ecosystem specifically involving the CRM-HL geometry has been established. The ecosystem resources include common, controlled CRM-HL geometry, specifications for consistent "reference" configurations, wind tunnel models designed and tested in multiple facilities, high-quality wind tunnel datasets, and computational models and simulation results, among others. The ecosystem will facilitate the cost sharing required for model development and testing, and enable the broad sharing of the generated test data with committed partners. A significant amount of wind tunnel data acquired through collaborative testing will be made publicly available. The ecosystem is expected to provide the following key benefits:

- Allows for the assessment of the accuracy, efficiency, and robustness of current and future CFD tools and technologies on industry-relevant configuration(s).
- Provides a common dataset (geometry and test data) that enables direct assessment and comparison between CFD flow solvers and modeling approaches.
- Provides a challenging high-lift geometry to assess the predictive capabilities of emerging computational tools, primarily through open community workshops, employing a broad array of test data (some of which will be held back to facilitate "blind" comparisons).
- With proper controls, enables the design and fabrication of nearly identical models in multiple facilities, which will help establish data repeatability, as well as explore issues such as wind tunnel testing uncertainties and scale effects.
- Provides a geometrically-complex configuration to demonstrate advanced flow measurement and sensing techniques to collect new, critical data needed for CFD model validation and development, and energizes new and continuing efforts within academia, hardware vendors, and other collaborators to develop such methods.
- Supports research activities in low-speed aerodynamic design, performance enhancement, noise reduction, and/or high-lift system simplification (e.g., slat/flap studies, flow control)
- Provides a freely-shareable geometry, which enables new, and strengthens existing, partnerships to accelerate technology development.

One of the key advantages of the ecosystem will be the ability to collect and prioritize wind tunnel data acquisition requirements to accelerate progress in high-lift aerodynamics computational prediction. These requirements include the type of data desired (e.g., surface flow visualization, off-body velocity field), the type of configurations that provide the best opportunity to acquire that data (e.g., small versus large, semispan versus full-span), and testing facilities that provide the desired flow environments (e.g., atmospheric, pressurized, or cryogenic). Established CRM-HL reference configurations will tie the wind tunnel models to multiple testing facilities throughout the ecosystem in a consistent manner, providing a clear research focus, and establishing conventional high-lift system performance levels for comparison.

An initial set of CRM-HL model development and testing campaigns has been committed. To date, NASA has completed the design and build of a 10% scale semispan model, and tested the model in their 14- by 22-Foot Subsonic Tunnel at the Langley Research Center in 2018 [21, 22]. That same model was subsequently installed and tested at the QinetiQ 5-metre (Q5m) Wind Tunnel in Farnborough, UK in 2019. The NASA test, primarily intended for exploring flow control, also acquired data on a baseline landing configuration without flow control. The Q5m test greatly expanded baseline testing by acquiring force/moment, pressure, and surface flow (tuft, oil-flow) data on both landing and takeoff configurations at different Reynolds numbers to explore optimal leading-edge (slat) and trailing-edge (flap) device positioning and sensitivities to inform the definition of the reference configuration. Currently, efforts are underway to design both a 5.2% scale semispan model and a 2.7% scale full-span model for testing at the NASA National Transonic Facility (NTF) in the 2021-2022 time frame. This testing will specifically explore Reynolds number and tunnel installation effects, as well as serve as a test-bed for innovative off-body velocity flow measurement techniques. In addition, a 6% scale full-span model will be developed and tested in the pressurized environment at the Q5m facility in 2022. That testing is expected to provide a wealth of data focusing on configuration variations, mounting system effects (like tare and interference), wall effects, and flow physics, all tying back to NTF-derived trends in Reynolds number. Additionally, a 4% scale semispan model will be designed and tested in multiple UK academic facilities to serve as a platform for UK aerodynamics research. Several other models are expected to be developed and tested in facilities in France, Germany, and Japan.

There are areas of critical interest for CFD modeling improvements that are not well-served by experimental studies on full configurations like the CRM-HL. In these instances, new canonical experiments could be proposed. One recent example of this type of test campaign is the NASA Juncture Flow (JF) experiment [23]. The Drag Prediction Workshop (DPW) series [24-27] identified the specific issue of CFD's inability to accurately and consistently predict wing-body-juncture separation. A new simplified experiment was designed to focus on this one issue, with the specific goal of measuring the junction flowfield details for the purpose of CFD validation.

Additionally, in areas where physical data are needed to better understand the nature of a particular flow, it is desirable to obtain measurements using two (or more) independent methods to help quantify measurement uncertainty. For example, as part of the JF series of experiments, flowfield velocity measurements are being obtained using two different laser-based techniques: laser Doppler velocimetry (LDV) and particle image velocimetry (PIV). These results will be compared with each other and documented in future publications.

4. Aerodynamic Aspects of Greatest Interest

Flow phenomena believed to have the biggest impact in high-lift flows are detailed below and are intended to highlight areas to focus attention when planning and conducting future wind tunnel tests. This is not intended to be a comprehensive list; focus areas are likely to be updated as new key lessons emerge.

It should also be noted that these key areas primarily focus on the high-lift aerodynamic performance of the CRM-HL. There are other related disciplines that are expected to find use cases for this geometry, and in some cases, the models that are being developed. Within AIAA, an initial stability and control (S&C) prediction workshop is planned around the high-speed CRM geometry. It is expected that future iterations of the workshop will make use of the low-speed CRM-HL models and testing being planned. In order for this to be successful, a unique set of requirements needs to be defined targeting S&C specific challenges. Similarly, the propulsion, noise, and icing communities are encouraged to identify use cases and requirements for specific testing that can be used for validation. From the propulsion community, it is expected that there will be a push for advancing physical testing techniques involving powered nacelles in a wind tunnel, for example.

1. Flow features generated by leading edge slat brackets

The nature of the flow behind each leading edge slat bracket on a typical high-lift configuration is very complex, and includes important flow features that are generated from relatively small geometric details on the bracket and wing surface. Figure 6 shows a typical slat element (transparent cyan), slat bracket (orange), and its associated structural cutout on the wing (blue) as modeled for CFD.

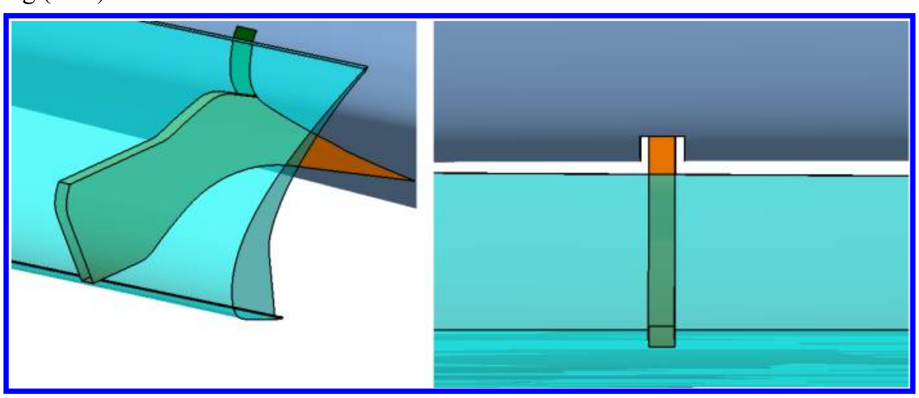

Figure 6: Typical Slat Bracket Geometry Modeling for CFD

The flow around slat bracket configurations leads to downstream flow separation that can often influence large scale flow characteristics on the wing itself. An example of this behavior can be seen in a view of the upper surface of the outboard CRM-HL wing close to the wingtip, as depicted in Figure 7. Oil flow emanating from the wing leading edge (top, not visible) illustrates the development of several separated flow wedges near the wing trailing edge (bottom) that are positioned directly behind the slat brackets. The flowfield in the vicinity of the slat brackets is complex, and is subject to both streamwise and spanwise flow, which lead to the formation of several distinct vortices along with low momentum flow (wake). As these features progress downstream and pass over the main wing's upper surface, they can trigger flow separation.

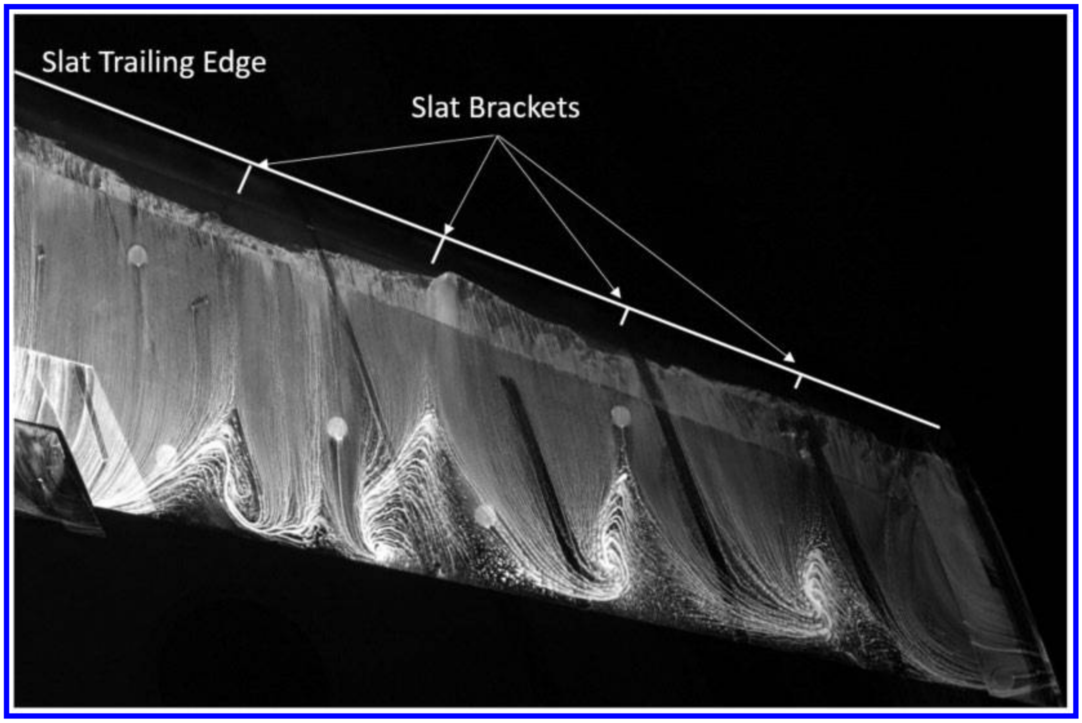

Figure 7: Oil Flow on the CRM-HL Outboard Wing

These areas of separated flow will impact performance and are important to properly characterize computationally. Computational results from previous high-lift workshops have shown that RANS methods struggle to properly predict flow separation growth in these areas, and often predict wedges of separated flow that are far too large relative to flow patterns seen in wind tunnel surface flow visualizations. Turbulence-resolving methods are showing more promise in these regions, but these methods require additional validation. A better understanding of vortex formation, convection, and growth near the slat brackets is seen as critical in validating and improving CFD modeling. Robust and efficient data collection techniques that are capable of accurately measuring off-body flow features starting from just downstream of the slat brackets and progressing downstream are critically needed.

2. Extent and nature of separation near $C_{L, max}$

In previous high-lift workshops, the emphasis was on modeling the correct lift levels and angle-of-attack at $C_{L, max}$. However, as mentioned earlier, it is often possible for CFD to obtain the correct lift levels near $C_{L, max}$ for the wrong reasons. In order to have confidence that the correct stall mechanism is being captured properly, details of the solutions at and beyond $C_{L, max}$ need to be compared to wind tunnel data. A particularly revealing parameter that is useful to consider when comparing CFD to wind tunnel data is pitching moment. However, even when matching lift and pitching moment, fortuitous error cancellation can still lead to results that seem reasonable but do not capture the physical mechanisms dominating the stall behavior. In addition to lift and pitching moment, surface flow visualization is instrumental in identifying key areas on the wing that break down first. Surface flow visualization, typically done using minitufts or oil flow, should be considered a minimum requirement for validating $C_{L, max}$. Examples of both are shown in Figures 8 and 9.

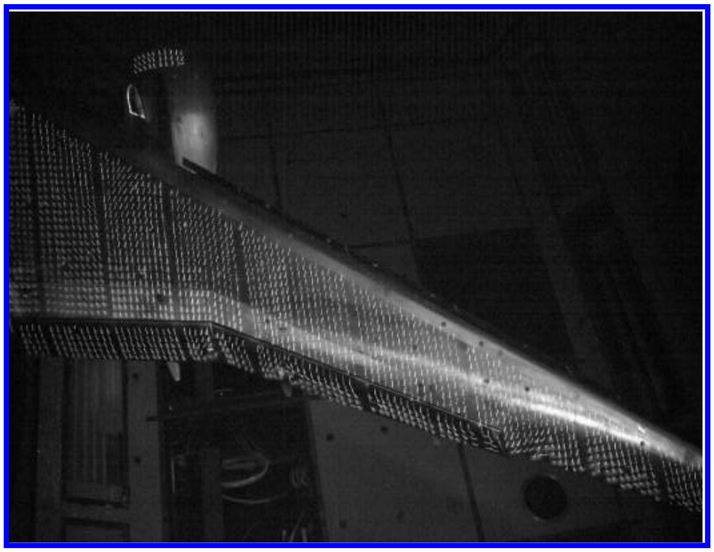

Figure 8: Minitufts on the CRM-HL at QinetiQ

Figure 9: Oil Flow on the JAXA Standard Model

Although surface flow visualization can be used to help identify areas where the loss of lift associated with $C_{L,\,max}$ occurs, it must be used in conjunction with the force and moment assessment to determine if a given simulation is correct or not. Wind tunnel testing of the CRM-HL 10% scale model in QinetiQ demonstrated three distinct areas of flow separation that can limit $C_{L,\,max}$, which depended specifically on the geometric configuration and the choice of wind tunnel floor boundary layer treatment. The first region that impacts stall is where flow from the nacelle and pylon spreads over the inboard wing, creating complex interactions of nacelle/pylon and chine vortex flow with the outboard end of the inboard slat. The second region that heavily influences the stall mechanism is where the wing root intersects the fuselage. Here, turbulent flow emanating from the inboard end of the inboard slat, complex flow around the wing strake, and a strong vortex from the fuselage body at high angles of attack all interact to create significant areas of flow separation on the inboard wing. For semispan ("half-model") configurations, the location and strength of the fuselage vortex is suspected to be strongly influenced by wind tunnel floor boundary layer and fuselage interactions. The third area that significantly impacts stall involves flow separation regions that occur over the outboard wing downstream of slat bracket wakes, as described above. The flow features in each of these three areas are important to model correctly in CFD as they directly influence $C_{L,\,max}$. To date, only a minimal amount of physical data exist in these areas to be useful for systematic CFD validation efforts.

3. Characterizing the nature of unsteady flow – frequencies and magnitudes

By design, RANS is intended to provide a time-averaged solution. It does not resolve any of the turbulent eddies in the flow, but rather models them, typically with linear or nonlinear eddy viscosity type turbulence models. Unsteady

RANS (URANS) can provide time-accurate results for large scale flowfield unsteadiness not directly related to turbulent eddy motions, but still relies on the turbulence modeling assumptions of the RANS formulation. As a result, URANS inherits all of the RANS deficiencies for modeling turbulence. For instance, URANS produces vortex shedding from a circular cylinder, but the mean lift and drag values are inaccurate. Most modern wind tunnels have developed various techniques to provide a single "averaged" data point (e.g., lift, drag) per test point, processed from the aerodynamic fluctuations that are physically present. This is generally an accepted practice, as the aerodynamic design processes that digest these results have evolved around using data in that format. Nonetheless, for highly separated flows near maximum lift, and for transient problems such as buffet, time-accurate on- and off-surface wind tunnel data (including balance forces and frequency data) are needed for validation of the unsteady flows that may be present.

RANS and URANS are generally trusted to make reasonable predictions for aerodynamic flows that do not involve significant regions of separation. For separated flows, the mean effects of the turbulence provided by these methods are not considered to be accurate enough [1]. By contrast, time-accurate scale-resolving methods (that resolve at least the larger turbulent eddies) are beginning to show promise toward reliably predicting separated flows (e.g., [17, 28, 29]). However, these methods are significantly more computationally expensive than RANS and URANS, so progress toward validating a robust, predictive capability for full-configuration high-lift configurations has been slow. As scale-resolving techniques become more widely available and applied to high-lift prediction, additional data requirements from physical testing will be required for CFD validation. Modern scale-resolving methods resolve at least the lower frequencies of turbulence spectra, so unsteady physical data such as unsteady surface pressures and time-accurate balance forces are necessary for validation of these CFD techniques.

4. Nacelle chine vortex characterization and its effects on the wing

A common feature on most modern commercial transport aircraft is a nacelle chine. The chine is a geometric extension like a vortex generator that is attached to the side of the nacelle, above its midline. It is positioned to have negligible aerodynamic impact during typical cruise conditions, but generates a strong vortex over the inboard wing at high aircraft angles-of-attack. At those conditions, the vortex from the chine interacts with the airflow over the inboard wing and increases performance, typically measured as higher lift levels or angle of attack at stall.

Current RANS methods can successfully capture the aerodynamic effects of the nacelle chine, such as the very strong vortex that is convected over the wing. However, depending on many factors including mesh resolution, turbulence modeling, and solver choices, the strength, location, and size of the predicted vortex core can vary. As a result, the interaction of the vortex flow feature with the wing boundary layer is likewise impacted. Often the only wind tunnel data to compare against are flow visualization or discrete pressure distributions on a limited area of the wing surface. If the computational and physical results do not agree, it becomes difficult to assess whether the differences are due to inaccurate modeling that occurs upstream of the wing leading edge, or to an inadequately captured nacelle chine vortex. Since the effect of the chine vortex is integral to high-lift performance, its strength and location need to be well quantified in the wind tunnel to increase confidence in predictions. Nacelle chine vortex flow characterization from initial generation through to its interactions with the wing upper surface is a high priority target for obtaining critical off-body flow measurements for CFD validation.

5. Reynolds number effects

A common challenge when evaluating high-lift performance on transport aircraft from data obtained within a wind tunnel environment is the effects of airplane scale, or Reynolds number. Typical commercially-utilized wind tunnels are sized to handle high-lift configuration models up to 10% in scale and up to 3 atmospheres in pressure. This gives a Reynolds number of only 15-30% relative to flight scale. Testing models at a significantly lower Reynolds number may lead to nonlinearities in performance that are often only discovered in flight tests. One alternative is to perform testing at cryogenic facilities that are capable of achieving near flight Reynolds numbers as well as independently varying both the dynamic pressure and Reynolds number. This allows the aerodynamic assessment over a wide range of Reynolds numbers, as well as providing an opportunity to isolate aeroelastic impacts from tunnel dynamic pressure. However, this comes at both increased operating cost and at significantly slower operational efficiencies.

Quantifying Reynolds number effects is an ideal use of CFD, where potential inconsistencies between flight and wind tunnel scale data could be discovered computationally well in advance of flight testing. This, however, requires CFD tools that are validated to adequately predict the effects of Reynolds number on aerodynamic performance. It is desired to test the CRM-HL configuration in cryogenic facilities ranging from low Reynolds numbers through flight scale. Key data from these tests would be used to identify and characterize flow features or performance discrepancies seen between Reynolds numbers, ideally using corroborating data obtained in multiple facilities using the same wind tunnel model. Comparisons could then be made between physical and computational results, in an effort to validate CFD's ability to predict the Reynolds number effects.

6. Boundary layer transition

Knowing the state of the boundary layer on the various elements of a high-lift configuration during a wind tunnel test is critical in correlating CFD simulation results with wind tunnel data. In practice, the exact state of the boundary layer is not often measured. Instead, common practice is to trip the boundary layer at a fixed location close to the leading edge of wing elements that are unlikely to otherwise naturally transition consistently. This is done to improve run-to-run repeatability by forcing transition to occur at the same place regardless of configuration or model condition. Forced tripping can be modeled in RANS by "turning off" the turbulence model in specified regions of the flow. However, this method can be tedious and unreliable. Therefore, fully turbulent CFD is often used to compare against tripped wind tunnel data, with small differences between tripping activation locations assumed to be small enough to ignore. In any case, the effectiveness of forced tripping in the wind tunnel should always be verified (for instance, using methods like infrared thermography).

When a wind tunnel model is tested "untripped" (without boundary layer trips, e.g., grit, trips strips or trip dots), it is important to measure where the transition from laminar to turbulent flow occurs. Because wind tunnel testing occurs at lower Reynolds numbers relative to flight, the boundary layers on the model will be thicker and generally transition to fully turbulent flow further downstream than on a similar flight vehicle configuration. As a result, the widely-used CFD modeling assumption of fully turbulent flow on each element of the model typically may not reflect the actual state of the boundary layer on a wind tunnel model. A subset of RANS turbulence models, generally known as "transition models," offers the capability to predict boundary layer transition. In some cases, it has been shown that capturing the effects of transition can influence the flow solution approaching $C_{L,\,max}$ [30, 31]. Many types of scale-resolving simulations may also be able to predict boundary layer transition, although current computational resource requirements are likely too high for their routine use on realistic configurations. Ultimately, detailed knowledge of the actual transition locations from the wind tunnel can be used to validate and improve both RANS-based transition models as well as scale-resolving methods. Primary regions of interest include, but are not limited to, the wing, slat, and flap upper surfaces and the nacelle lip.

7. Flap separation & increments vs deflection

One important aerodynamic aspect that has not received adequate attention in the HLPW series to date is trailing edge flap performance. Changes in flap deflection typically alter lift performance in the operational angle-of-attack range, while having only secondary impacts near $C_{L,\,max}$. Maximum loading on a trailing edge flap typically occurs in this lower angle-of-attack range, making it more challenging to predict computationally compared to when the flap becomes unloaded near maximum airplane lift. Also, most design work, and therefore most needed CFD validation, is performed in this lower angle-of-attack range.

Two off-nominal landing deflections for the trailing edge flaps are planned to be released with the reference CRM-HL geometry. These will represent +3° and -3° of deflection relative to the nominal flap position for both the inboard and outboard flaps. Community assessment of CFD for increments between these deflections are also featured in the next HLPW planned for 2021. Requirements for validation are similar to what is required for other areas, namely a better understanding of flow phenomena that drive performance, particularly if there are significant differences observed between deflections in the CFD results. Of primary interest are surface pressures and surface flow visualization of the flaps for an aircraft angle of attack of approximately 6°.

8. Component buildup

A key feature of the CRM-HL wind tunnel model architectural design will be the ability to test a series of increasingly complex, modular geometric configurations, allowing aerodynamic impacts of particular components to be isolated. Potential modular components include the wing, the leading edge devices, the trailing edge devices, the nacelle, the chine, and landing gear. The objective of this approach is to complement potential isolated canonical flows, by acquiring test data on simpler configurations where CFD can be more readily validated. This allows a building-block assessment of key flow features to better understand the CFD predictions on full configurations. If CFD cannot accurately model the flow features for simpler configurations, then it is unlikely to be able do so for the more complex configurations. Validation would logically start from the simplest configurations, then add complexity as best practices are developed.

The simplest typical configuration of regular interest is the wing-body configuration. In this configuration, the wing would have no leading or trailing edge devices deployed, nor contain the installed nacelle-pylon. In this very simple configuration, RANS would be expected to perform well before significant regions of separation are present, but potentially struggle as separation appears. If this is the case, then presumably such wind tunnel testing could yield beneficial validation data for scale-resolving method development. If attached predictions don't agree with wind tunnel testing in the wing-body configuration, an even simpler body only configuration could be explored. This could be useful for investigations into CFD modeling of the wind tunnel, or for analysis of much simpler aerodynamic flows. The body only configuration should be very easy to predict computationally, and any mispredictions could be investigated free of other potentially complex influences.

Advancing toward high lift, the next components that would be added are the leading edge devices. The slat and wing-under-slat-surface (WUSS) are lofted and positioned to be modeled as spanwise-continuous when the nacelle and pylon components are not installed. In this configuration, spanwise breaks are minimized, which results in a flowfield that is significantly less complex than the full high-lift configuration. Next, the cruise trailing edge would be replaced with the landing flap trailing edge. This results in the simplest high-lift configuration, but without the nacelle-pylon. Finally, adding in the nacelle-pylon brings the model to the baseline high-lift configuration.

Another variation of interest is the takeoff configuration. In this configuration, the trailing edge flap deflection is reduced to yield fully-attached flow throughout a larger angle-of-attack range. Additionally, the leading edge slats, which in the CRM-HL landing configuration have a gap that allows airflow between them and the main element, are sealed to the WUSS, blocking off this energizing flow. This configuration represents a typical commercial aircraft takeoff geometry, and additionally provides a configuration that minimizes the aerodynamic effects of the leading edge brackets. Furthermore, this configuration has a different mechanism driving stall breakdown compared to the landing configuration, and therefore, needs to be given carefully consideration alongside the landing configurations.

5. Wind Tunnel Related Requirements

In the context of the CRM-HL ecosystem, CFD validation will primarily involve comparing computational results with physical measurements obtained during wind tunnel testing. The conclusiveness of the outcomes of such comparisons will depend on a number of factors, including the ways in which both wind tunnel models and wind tunnel test campaigns are designed.

In particular, preliminary computational studies are envisioned to guide effective wind tunnel testing in support of high-lift CFD validation. Such studies will help inform decision making and identify priorities for wind tunnel test campaigns. It is vital that preliminary computational and wind tunnel-based studies are coordinated in a balanced and synergistic way to maximize mutual benefit. We refer to this synergy as a "validation dialog." This type of collaborative effort is exemplified by the NASA Juncture Flow experiment [32, 33]. Establishing an effective validation dialog not only accelerates the pace of learning with regard to CFD, but the iterative nature of performing the coordinated studies also provides additional insights to the broader high-lift community. There are several aspects of contemporary high-lift wind tunnel testing practice that will benefit from the greater scrutiny that such collaborative studies will afford, some of which are outlined below.

One area for additional study is the current practice of applying primary corrections for the effects of wall and support interference. It is known that both effects possess three-dimensional gradients in the vicinity of high-lift models [34]. The additional insight afforded by CFD computations of the wind tunnel environment, with and without a model, may be expected to lead to an improved understanding of wall and support interference effects – potentially leading to improved methods for developing the associated corrections. Obtaining better and more definitive data to quantify these effects provides a real opportunity in the high-lift community to demonstrate validation dialog and work together to improve high-lift testing.

To advance wind tunnel testing effectiveness for high-lift CFD validation, we must develop specific requirements associated with the representation of the wind tunnel environment in CFD computations, and measurement of physical properties during the wind tunnel test [35]. In either case, there is not a definitive (robust, evidence-based and realizable) set of requirements that currently exists for high-lift CFD validation. Consequently, the material presented below outlines some guidelines to aid in the establishment of such requirements.

1. Representation of the wind tunnel environment in CFD computations

In principle, it is important to represent the physical set-up in the wind tunnel as closely as possible in CFD simulations in order to reduce uncertainties in interpreting differences between computed results and physical measurements. A principal focus is in establishing clear requirements for how the boundary conditions used in CFD simulations should be defined. Boundary treatments are required for both the geometry (location, shape) and for the thermodynamic properties (pressure, temperature, velocity, etc.). In both respects, key parameters are required as either inputs or constraints in any CFD simulation. From the perspective of the CFD practitioner, it is clearly desirable that all wind tunnel boundary conditions are either known (a priori) or knowable (a posteriori). However, the extent to which either of these states are attainable in practice remains to be seen. Some of the challenges, and opportunities for the CRM-HL ecosystem, are outlined below.

Geometry

Those involved in CFD validation typically request as many geometric details as possible of the wind tunnel test environment. These details typically include the windswept profiles of the facility walls, running from the start of the contraction through the diffuser, together with the OML and positioning of the as-built model and its support hardware in the test section. Wind tunnel operators appreciate the importance of these features [36, 37]. However, it is not well understood what level of detail or geometric accuracy is required in order to facilitate a desired level of high-lift CFD modeling accuracy.

Determining whether or how to include a specific geometric feature in a CFD simulation will require focused efforts beyond just wind tunnel testing a fully-featured CRM-HL model. To this point, suitably designed CFD studies can often yield valuable insights into what is important (or possible) to include when defining geometry. CFD-based sensitivity studies can be conducted at usually much less cost and with considerably less constraints in terms of acquiring test facility access. Data from such studies are beginning to emerge [35, 38] and will help to provide justification and establish priorities throughout the CRM-HL ecosystem.

Thermodynamics

As with the geometric details, the thermodynamic properties over the geometric boundaries of a CFD simulation should be known and properly defined. Generally speaking, the challenges associated with characterizing the upstream and downstream flow conditions are not decoupled from those associated with defining the streamwise extent of the wind tunnel to be included in the computations. Because there is still much to be learned concerning what flow quantities to measure and where, we suggest establishing validation dialog between CFD modeling and wind tunnel testing experts in this area. Moreover, rather than considering the most demanding aspects to define initially, a systematic approach should be taken, breaking large complex problems down into smaller, more focused activities where necessary. Two such areas of focus include:

1. Understanding the requirements associated with prescribing and measuring upstream and downstream boundary conditions. This will need to consider spatial and, potentially, temporal nonuniformities, in the plane of and normal to each boundary. Data should be collected with the model in situ and compared to empty test section calibration data (when available) to establish the sensitivity, if any, of boundary conditions to the model being evaluated. It may also be necessary to represent events occurring upstream of the upstream boundary, perhaps associated with the action of the main drive fan, for instance. The upstream transmission of disturbances originating further downstream may also need to be considered. All such representations will likely prove to be wind tunnel facility dependent.
2. Understanding how to better represent model support effects, especially those associated with semispan model testing. This type of testing is often favored for high lift since larger models may be used, thereby increasing the Reynolds numbers (based on mean aerodynamic chord) and increasing the amount of instrumentation that can be accommodated within the model. However, with semispan testing, the model is no longer immersed in the more uniform onset flow toward the center of the test section. Rather, a semispan model is partially immersed in the boundary layer associated with the wall on which it is mounted. A variety of techniques, like using a stand-off to raise the model beyond the immediate influence of the undisturbed boundary layer, have been used to reduce the impact on the flow over the half-body. There is limited consensus regarding the effectiveness of such techniques in the presence of the large local flow gradients evident at high lift, especially given the inclination of the partially immersed fuselage to the oncoming boundary layer at moderate angles of attack.

Both of the above topics lead to the recommendation of conducting coordinated campaigns across several wind tunnels, using both full-span and semispan models. Planning for this is already underway within the CRM-HL eco-system.

2. Measurement of physical properties during wind tunnel testing

Wind tunnel test activities specifically undertaken for CFD validation purposes will generally differ from those employed to support more conventional high-lift characterization activities, both in terms of their manner of conduct and the measurement data acquired. For instance, in order to allow the influence of certain aspects of the test environment to be characterized numerically, it may be necessary to consider using installation arrangements that have little direct practical interest from the perspective of establishing high-lift performance: for example, placing the model in a nonstandard location in the test section, or varying key tunnel operating parameters (such as sidewall blowing or suction levels) beyond their normal limits. The use of high-lift device rigging settings that have little or no practical significance might also be considered. Tunnel test campaigns could also be designed to ensure that the sequence in which tests are conducted and the way in which measurements are taken supports a rigorous statistical analysis of the data.

The scope and detail of the measurements to be made are going to be far more substantial than those required to support more routine wind tunnel operation. The data required to facilitate accurate definition of CFD boundary conditions, discussed above, are likely to be rather different than that gathered for traditional purposes (e.g., to verify that the desired reference conditions have been established in the test section, or to derive primary corrections for wall and support interference or flow angularity). The scope and detail (both spatial and temporal) of on- and off-surface data requirements are also likely to be expanded considerably (e.g., detailed quantitative skin-friction measurements, or PIV close to the surface). In turn, this may have important consequences for the design of the wind tunnel model. Modifications may also need to be made to the installation equipment and even the wind tunnel facility itself in order to provide optical access or accommodate instrumentation (such as sensors or in situ data storage/processing hardware and supporting communication/transmission devices).

Another key element pertains to the establishment of instrumentation requirements. Whether the tests are intended to characterize the flow about a fully-configured high-lift system or are focused on more localized or isolated flow phenomena, there does not appear to be a robust or independent set of a priori criteria on the type, size, or application of instrumentation needed to acquire accurate on-surface or off-body data. Moreover, despite the wealth of literature on the subject (Ref. [39]), practical guidance on how such criteria should be established is not readily available.

Setting aside its current limitations, CFD has a vital role to play in developing requirements in the measurement of physical properties [20]. Generally speaking, wind tunnel tests intended to support CFD validation activities should not be planned without first conducting systematic studies of the flows of interest using CFD. Given that the spatial and temporal resolution of physical measurements are likely to be more constrained than the data generated with CFD, deliberate and thoughtful pretest CFD studies will help prioritize measurement requirements. In view of the relative difficulty and expense associated with characterizing the flow in wind tunnel facilities, such preparatory activities should include simplified CFD-based sensitivity studies to address the potential impact of any anticipated deficiencies in the planned physical measurements.

All of the above elements serve to illustrate that various trade-offs will need to be made in order to meet effective test objectives. In view of the diversity and scope of the requirements to support high-lift CFD validation, the task of establishing and prioritizing wind tunnel test objectives is challenging. Thus, the importance of establishing and maintaining a vibrant validation dialog between the CFD and wind tunnel testing communities cannot be overstated here. A balance between gathering the physical data required to support CFD validation and using CFD to help establish the requirements for the physical data to be gathered will need to be determined. As a result, it is recommended that extensive CRM-HL CFD studies be undertaken alongside the conventional wind tunnel test campaigns that are currently being coordinated for a suite of full- and semispan models in a number of wind tunnel facilities.

6. Data Accuracy and Uncertainty Quantification

A crucial and often overlooked aspect of any CFD validation activity is an assessment of the demonstrated “fitness for purpose” or “suitability for intended use” [12, 40, 41]. A sound appreciation of the underlying requirements for predictive accuracy is fundamental here. However, there is currently no open or agreed framework defining high-lift CFD prediction data accuracy requirements that may be used to drive CRM-HL ecosystem endeavors. The establishment of such a framework may require considerable effort. In the past, a concerted international activity, coordinated under the auspices of NATO, was undertaken in order to establish such requirements for wind tunnel data accuracy [42]. It is hoped that the CRM-HL ecosystem will provide the platform and impetus for the establishment of such a framework. In the meantime, without uncertainty bounds, or even a practical method of determining uncertainty bounds, an open question then is “how close is close enough?” To answer that, it is useful to consider what incremental differences in aerodynamic parameters may be acceptable in industrial practice. From this perspective, differences of in 0.03 in $C_{L,\,max}$, or ten drag counts (0.0010 in C_D) are considered meaningful performance metrics for transport aircraft in a high-lift configuration.

Whatever accuracy requirements are employed during CFD validation assessment, the quantification of the uncertainty present in both the computed CFD data and the physical measurements from wind tunnel testing plays a crucial role. Without this, the confidence in any validation assessment is reduced by difficulties in properly attributing the uncertainties to test and/or CFD data. Conversely, if uncertainties aren't properly communicated, recipients of data may assume more accuracy then there may actually be. For instance, differences between results from CFD and wind tunnel measurements may be misconstrued to be primarily a consequence of deficiencies in the CFD model, when in fact the uncertainty bounds in the CFD results may lie within, or at least overlap with, the wind tunnel data uncertainty bounds.

One of the greatest challenges pertinent to both CFD and wind tunnel testing communities is balancing the justification for obtaining the additional data required to support formal uncertainty quantification (UQ) with the cost and practicality of its acquisition. The additional requirements here impact every conceivable aspect of the computations, and physical measurement acquisition from model manufacturing imperfections to establishment of the test section inflow conditions. This is a field of considerable research interest in both communities [43, 44]. Even if there is currently not a consistent methodology for tying everything together, more information can often lead to an improved understanding of the level of variations that may be observed under different scenarios.

The CRM-HL ecosystem can play an important role in supporting the development of improved UQ methodologies by (i) allowing disparate local practices, CFD tools, and wind tunnel facilities to be exercised and compared using a common, shared basis: the CRM-HL, and (ii) by providing the incentive to gather, systematically, data in greater detail than might otherwise be possible to justify. For instance, the importance and relative economy of coordinated CFD-based preparatory studies has been discussed in sections 2 and 4, and the benefits of testing the same model in different wind tunnels in Section 5. A further specific example would be to test the planned 6% full-span model under development not only at the QinetiQ 5-metre facility, but also at a comparable facility such as the ONERA F1 tunnel. Comparing the two wind tunnel datasets may shed light on tunnel wall interference effects on aerodynamic performance, effects of mounting systems, potential differences in wall correction methodologies, etc. Conversely, there is value in testing similar, but different models developed among ecosystem partners in the same facility.

7. Closing Remarks

The challenges associated with CFD validation for a transport aircraft in a high-lift configuration identified in the preceding sections are numerous, and often do not have clear paths toward resolution. It is hoped that identification of these areas and the indication of their importance will drive further research, both computationally and physically, and help to refine requirements that can begin to address them. It is fully expected that the requirements will continue to evolve as our understanding of high-lift flow increases and CFD technology continues to improve. Some requirements may include targeted measurements, others may lead to the identification of additional canonical flow cases, or new use cases for the ecosystem.

The CRM-HL ecosystem has been devised to provide a powerful vehicle to support such endeavors, in ways that would otherwise not be possible in the public domain, and could not be afforded in the private sector. After many years of planning, the CRM-HL ecosystem is now gaining momentum, with new wind tunnel models being commissioned and several wind tunnel test campaigns in their early planning stages. Measurements beyond those typically required for configuration development will be necessary to support in-depth studies of both the facilities themselves, as well as specific flow features on the models. Similarly, CFD practitioners will need to conduct detailed studies aimed at addressing the same challenges. In addition to ensuring that all CFD codes employed for the effort have undergone rigorous code verification, efficient solution verification methods need to be developed for emerging methods to support these studies. Significant pretest studies will also need to be conducted to support the development of test plans. For the goals of the CRM-HL ecosystem to be realized, the teams responsible for wind tunnel tests and CFD computations are working in tandem to develop logical and well-organized plans to address the current challenges, and to collectively advance the state of the art in high-lift aerodynamics.

References

[1] Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., and Mavriplis, D., "CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences, NASA/CR-2014-218178, March 2014; https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20140003093.pdf.

[2] Lacy, D. S. and Sclafani, A. J., "Development of the High Lift Common Research Model (HL-CRM): A Representative High Lift Configuration for Transonic Transports," AIAA Paper 2016-0308, January 2016; doi: https://doi.org/10.2514/6.2016-0308.

[3] "Recommended Practices When Flight Modeling is Used to Reduce Flight Testing Supporting Aircraft Certification," AIAA Standards Document (To be published)

[4] Slotnick, J. P., Hannon, J. A., and Chaffin, M., "Overview of the First AIAA CFD High Lift Prediction Workshop (Invited)," AIAA Paper 2011-0862, January 2011; doi: https://doi.org/10.2514/6.2011-862.

[5] Rumsey, C. L., Slotnick, J. P., Long, M., Stuever, R. A., and Wayman, T. R., "Summary of the First AIAA CFD High-Lift Prediction Workshop," Journal of Aircraft, Vol. 48, No. 6, 2011, pp. 2068-2079, doi: https://doi.org/10.2514/1.C031447.

[6] Johnson, P. L., Jones, K. M., and Madson, M. D., "Experimental Investigation of a Simplified 3D High Lift Configuration in Support of CFD Validation," AIAA Paper 2000-4217, August 2000, doi: https://doi.org/10.2514/6.2000-4217.

[7] Hannon, J. A., Washburn, A. E., Jenkins, L. N., and Watson, R. D., "Trapezoidal Wing Experimental Repeatability and Velocity Profiles in the 14- by 22-Foot Subsonic Tunnel (Invited)," AIAA Paper 2012-0706, January 2012, doi: https://doi.org/10.2514/6.2012-706.

[8] Rumsey, C. L. and Slotnick, J. P., "Overview and Summary of the Second AIAA High-Lift Prediction Workshop," Journal of Aircraft, Vol. 52, No. 4, 2015, pp. 1006-1025, doi: https://doi.org/10.2514/1.C032864.

[9] Rudnik, R., Huber, K., and Melber-Wilkending, S., "EUROLIFT Test Case Description for the 2nd High Lift Prediction Workshop," AIAA Paper 2012-2924, June 2012, doi: https://doi.org/10.2514/6.2012-2924.

[10] Rumsey, C. L., Slotnick, J. P., and Sclafani, A. J., "Overview and Summary of the Third AIAA High-Lift Prediction Workshop," Journal of Aircraft, Vol. 56, No. 2, 2019, pp. 621-644, doi: https://doi.org/10.2514/1.C034940.

[11] Yokokawa, Y., Murayama, M., Uchida, H., Tanaka, K., Ito, T., and Yamamoto, K., "Aerodynamic Influence of a Half-Span Model Installation for High-Lift Configuration Experiment," AIAA Paper 2010-0684, January, 2010, doi: https://doi.org/10.2514/6.2010-684.

[12] Oberkampf, W. L. and Roy, C. J., Verification and Validation in Scientific Computing, Cambridge University Press, Cambridge, UK, 2010.

[13] Ollivier Gooch, C. F., "Is the Problem with the Mesh, the Turbulence Model, or the Solver? Statistical Analysis of High Lift and Drag Prediction Workshop Data," AIAA Paper 2019-1334, January 2019; doi: https://doi.org/10.2514/6.2019-1334.

[14] Spalart, P. R. and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," Recherche Aerospatiale, Vol. 1, 1994, pp. 5-21.

[15] Rumsey, C. L., "NASA Langley Turbulence Modeling Resource Website," https://turbmodels.larc.nasa.gov, accessed 2020-05-07.

[16] Ito, Y., Murayama, M., Yokokawa, Y., Yamamoto, K., Tanaka, K., Hirai, T., Yasuda, H., Tajima, A., and Ochi, A., "JAXA's and KHI's Contribution to the Third High Lift Prediction Workshop," Journal of Aircraft, Vol. 56, No. 3, 2019, pp. 1080-1098; doi: https://doi.org/10.2514/1.C035131.

[17] Konig, B., Fares, E., Murayama, M., and Ito, Y., "PowerFLOW Simulations for the Third AIAA High Lift Prediction Workshop." AIAA Paper 2018-1255, January 2018; doi: https://doi.org/10.2514/6.2018-1255.

[18] Mavriplis, D. J., Vassberg, J. C., Tinoco, E. N., Mani, M., Brodersen, O. p., Eisfeld, B., Wahls, R. A., Morrison, J. H., Zickuhr, T., Levy, D., and Murayama, M., "Grid Quality and Resolution Issues from the Drag Prediction Workshop Series," Journal of Aircraft, Vol. 46, No. 3, 2009, pp. 935-950; doi: https://doi.org/ 10.2514/1.39201.

[19] Taylor, N. et al., "The Prediction of Vortex Interactions on a Generic Missile Configuration Using CFD: Current Status of Activity in NATO AVT-316," NATO STO-MP-AVT-307, Paper 24, October 2019.

[20] Taylor, N., “Separated Flow: Some Challenges and Research Priorities for Missile Aerodynamics,” NATO STO-MP-AVT-307, Paper 23, October 2019
[21] Lin, J. C., Melton, L. P., Hannon, J. A., Andino, M. Y., Koklu, M., Paschal, K. B., Vatsa, V. N., “Wind Tunnel Testing of Active Flow Control on High-Lift Common Research Model, AIAA Paper 2019-3723, June 2019; doi: https://doi.org/10.2514/6.2019-3723.
[22] Koklu, M., Lin, J. C., Hannon, J. A., Melton, L. P., Andino, M. Y., Paschal, K. B., and Vatsa, V. N., “Surface Flow Visualization of the High-Lift Common Research Model,” AIAA Paper 2019-3727, June 2019; doi: https://doi.org/10.2514/6.2019-3727.
[23] Kegerise, M. A., Neuhart, D. H., “An Experimental Investigation of a Wing-Fuselage Junction Model in the NASA Langley 14- by 22-Foot Subsonic Tunnel,” NASA/TM-2019-220286, June 2019; https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20190027403.pdf.
[24] Tinoco, E. N., Brodersen, O. P, Keye, S., Laflin, K. R., Feltrop, E., Vassberg, J. C., Mani, M., Rider, B., Wahls, R. A., Morrison, J. H., Hue, D., Roy, C. J., Mavriplis, D., and Murayama, M., “Summary Data from the Sixth AIAA CFD Drag PredictionWorkshop: CRM Cases,” Journal of Aircraft, Vol. 55, No. 4, 2018, pp. 1352-1379, doi: https://doi.org/10.2514/1.C034409
[25] Levy, D., Laflin, K., Tinoco, E., Vassberg, J., Mani, M., Rider, B., Rumsey, C., Wahls, R., Morrison, J., Broderson, O., Crippa, S., Mavriplis, D., and Murayama, M., “Summary of Data from the Fifth Computational Fluid Dynamics Drag Prediction Workshop,” Journal of Aircraft, Vol. 51, No. 4, 2014, pp. 1194-1213, doi: https://doi.org/10.2514/1.C032389.
[26] Vassberg, J. C., Tinoco, E. N., Mani, M., Rider, B., Zickuhr, T., Levy, D. W., Brodersen, O. P., Eisfeld, B., Crippa, S., Wahls, R. A., Morrison, J. H., Mavriplis, D. J., and Murayama, M., “Summary of the Fourth AIAA Computational Fluid Dynamics Drag Prediction Workshop,” Journal of Aircraft, Vol. 51, No. 4, 2014, pp. 1070-1089; doi: https://doi.org/10.2514/1.C032418.
[27] Vassberg, J. C., Tinoco, E. N., Mani, M., Brodersen, O. P., Eisfeld, B., Wahls, R. A., Morrison, J. H., Zickuhr, T., Laflin, K. R., and Mavriplis, D. J., “Abridged Summary of the Third AIAA Computational Fluid Dynamics Drag Prediction Workshop,” Journal of Aircraft, Vol. 45, No. 3, 2008, pp. 781-798; doi: https://doi.org/10.2514/1.30572.
[28] Uzun, A. and Malik, M., “Wall-Resolved Large-Eddy Simulations of Transonic Shock-Induced Flow Separation,” AIAA Journal, Vol. 57, No. 5, 2019, pp. 1955-1972; doi: https://doi.org/10.2514/1.J057850.
[29] Iyer, P. S., Park, G. I., and Malik, M., “Wall-Modeled Large Eddy Simulation of Transonic Flow over an Axisymmetric Bump with Shock-Induced Separation,” AIAA Paper 2017-3953, June 2017; doi: https://doi.org/10.2514/6.2017-3953.
[30] Coder, J. G., “OVERFLOW Analysis of the DLR-F11 High-Lift Configuration Including Transition Modeling,” Journal of Aircraft, Vol. 52, No. 4, July–August 2015, pp. 1082-1097; doi: https://doi.org/10.2514/1.C033145.
[31] Langlois, M., Yang, H., and Sermeus, K., “Bombardier Contribution to the 3rd AIAA High-Lift Workshop,” AIAA Paper 2018-1038, January 2018; doi: https://doi.org/10.2514/6.2018-1038.
[32] Rumsey, C. L., “The NASA Juncture Flow Test as a Model for Effective CFD/Experimental Collaboration,” AIAA Paper 2018-3319, June 2018; doi: https://doi.org/10.2514/6.2018-3319.
[33] Rumsey, C. L., “Exp: NASA Juncture Flow (JF),” data provided on the NASA Langley TMR Website, https://turbmodels.larc.nasa.gov/Other_exp_Data/junctureflow_exp.html, accessed 2020-05-07.
[34] Ewald, B.F.R. (ed), “Wind Tunnel Wall Correction”, AGARD-AG-336, October 1998.
[35] Rumsey, C. L., Carlson, J.-R., Hannon, J. A., Jenkins, L. N., Bartram, S. M., Pulliam, T. H., and Lee, H. C., “Boundary Condition Study for the Juncture Flow Experiment in the NASA Langley 14x22-Foot Subsonic Wind Tunnel,” AIAA Paper 2017-4126, June 2017; doi: https://doi.org/10.2514/6.2017-4126.
[36] Jaarsma, F., “General Design Aspects of Low Speed Wind Tunnels”, AGARD-CP-585, Paper 3, June 1997.
[37] Woodward, D.S., Francois, G. and Taylor, N., “The Aerodynamic and Structural Design of the DRA 5-Metre and ONERA F1 Low-Speed Pressurised Wind Tunnels”, AGARD-CP-585, Paper 4, June 1997.
[38] Rudnik, R. and Melber-Wilkending, S., “CFD Concepts for In-Tunnel Simulation of Low Speed Wind Tunnels with Closed Test Sections,” AIAA Paper 2020-0380, January 2020; doi: https://doi.org/10.2514/6.2020-0380.

[39] Oberkampf, W. L. and Smith, B. L., "Assessment Criteria for Computational Fluid Dynamics Model Validation Experiments," Journal of Verification, Validation and Uncertainty Quantification, Vol. 2, 031002, pp. 1-14; doi: https://doi.org/10.1115/1.4037887.
[40] Taylor, N, "CFD Validation – What is it and how does it affect us?", RAeS Aerodynamics Conference, October 2019.
[41] Lee, H. B., Urmila, G, Sami, B, Oberkampf, W. L., Roy, C. J., Benek, J. A., Rumsey, C. L., Powers, J. M., Bush, R. H., Mani, M., "Development and Use of Engineering Standards for Computational Fluid Dynamics for Complex Aerospace Systems", AIAA Paper 2016-3811, June 2016; doi: https://doi.org/10.2514/6.2016-3811
[42] Steinle, F. and Stanewsky, E., "Wind Tunnel Flow Quality and Data Accuracy Requirements", AGARD-AR-184, November 1982.
[43] Schaefer, J. A., Romero, V. J., Schafer, S. R., Leyde, B., Denham, C. L., "Approaches for Quantifying Uncertainties in Computational Modeling for Aerospace Applications," AIAA Paper 2020-1520, January 2020; doi: https://doi.org/10.2514/6.2020-1520.
[44] Rhode, M. N. and Oberkampf, W. L., "Estimation of Uncertainties for a Model Validation Experiment in a Wind Tunnel," Journal of Spacecraft and Rockets, Vol. 54, No. 1, 2017, pp. 155-168; doi: https://doi.org/10.2514/1.A33563.

This article has been cited by:

1. Paul M. Danehy, Ross A. Burns, Daniel T. Reese, Jonathan E. Retter, Sean P. Kearney. 2022. FLEET Velocimetry for Aerodynamics. *Annual Review of Fluid Mechanics* **54**:1, 525-553. [Crossref]
2. Konrad Goc, Sanjeeb T. Bose, Parviz Moin. Large Eddy Simulation of the NASA High-Lift Common Research Model . [Abstract] [PDF] [PDF Plus]
3. Nigel J. Taylor. AVT-316 Missile Facet: Lessons learned concerning the prediction of Vortex Flow Interactions about Generic Missile Configurations . [Abstract] [PDF] [PDF Plus]
4. Gokul Subbian, Andrea Magrini, Denis Buosi, Rolf Radespiel, Rita Ponza, Ernesto Benini. Investigation of HL-CRM Aerodynamics With a UHBPR Nacelle in Powered-on Condition . [Abstract] [PDF] [PDF Plus]
5. David P. Lockard, Travis L. Turner, Christopher J. Bahr, Florence V. Hutcheson. Overview of Aeroacoustic Testing of the High-Lift Common Research Model . [Abstract] [PDF] [PDF Plus]
6. Daniel T. Reese, Ryan J. Thompson, Ross A. Burns, Paul M. Danehy. 2021. Application of femtosecond-laser tagging for unseeded velocimetry in a large-scale transonic cryogenic wind tunnel. *Experiments in Fluids* **62**:5. . [Crossref]
7. Jeffrey P. Slotnick, Dimitri Mavriplis. A Grand Challenge for the Advancement of Numerical Prediction of High Lift Aerodynamics . [Abstract] [PDF] [PDF Plus]
8. Nigel J. Taylor, Christopher L. Rumsey. CFD Validation Experiments: Toward a Broader Perspective . [Abstract] [PDF] [PDF Plus]
9. Konrad A. Goc, Oriol Lehmkuhl, George Ilhwan Park, Sanjeeb T. Bose, Parviz Moin. 2021. Large eddy simulation of aircraft at affordable cost: a milestone in computational fluid dynamics. *Flow* **1**. . [Crossref]
10. Ashley N. Evans, Doug S. Lacy, Ian Smith, Melissa B. Rivers. Test Summary of the NASA High-Lift Common Research Model Half-Span at QinetiQ 5-Metre Pressurized Low-Speed Wind Tunnel . [Abstract] [PDF] [PDF Plus]